

ECEN 150 Lab 12 – LTspice Introduction

Name: _____

Purposes: (34 points total)

- Learn the basics of using LTspice for circuit analysis.
- Review Ohm's Law and the maximum power transfer theorem.

Introduction

Large circuits cannot be easily solved by hand or evaluated on a breadboard, so computer simulations are widely used in the electronics industry. **SPICE** is an industry standard method for simulating analog circuits under DC, AC, and transient scenarios. **Simulation Program with Integrated Circuit Emphasis** is open source. The original was text-based. Today many enhanced variants of SPICE exist and include schematic editors and graphical user interfaces (GUIs). LTspice is a free version of SPICE with these added amenities.








Preparation

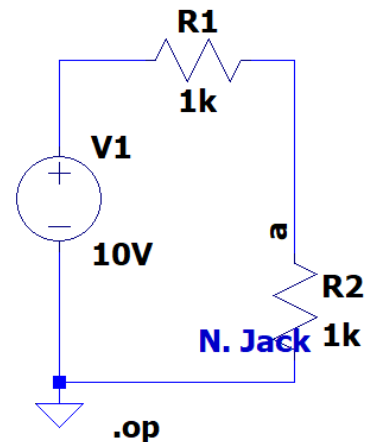
- Install **LTspice** from the Analog Devices website using the following link: [LTspice](#)
- *Mac users:
 - While LTspice exists for Mac, it can be difficult to use. For future engineering classes you should strongly consider installing Windows on your Mac. The BYUI Technology Repair center at the bookstore can assist you.
 - This video may be helpful as you try to finish this lab: [Mac LTspice tutorial](#)



Procedure:

Part 1. DC operating point simulation.


Step 1: Create the circuit.

- With LTspice running, create a new schematic from the menu **File → New Schematic**.
- Save the schematic: **File → Save as**. Give it a name and location and then save as a .asc file.
- Create the schematic shown here. (*your background color will look different)
 - Click **Edit → Component** to add components. (or press F2, or click )
 - “voltage” = DC voltage source.
 - “res” = resistor (or the toolbar icon ).
 - Values: right-click any value or name to edit them.
No spaces!
 - Rotation: “ctrl r” before placing or while moving
 - Moving after placement: Click  and then the element.
 - Wires: **Edit → Draw wire** (or toolbar icon ). Then click the two terminals to connect.
 - Ground: **Edit → Place GND** (or toolbar icon )
 - Label node “a”: **Edit → Label Net** (or icon )
 - Enter the name, click OK, then click on the node.
 - Text: **Edit → Text** (or ). Enter **your name**, OK, then place it on the schematic.

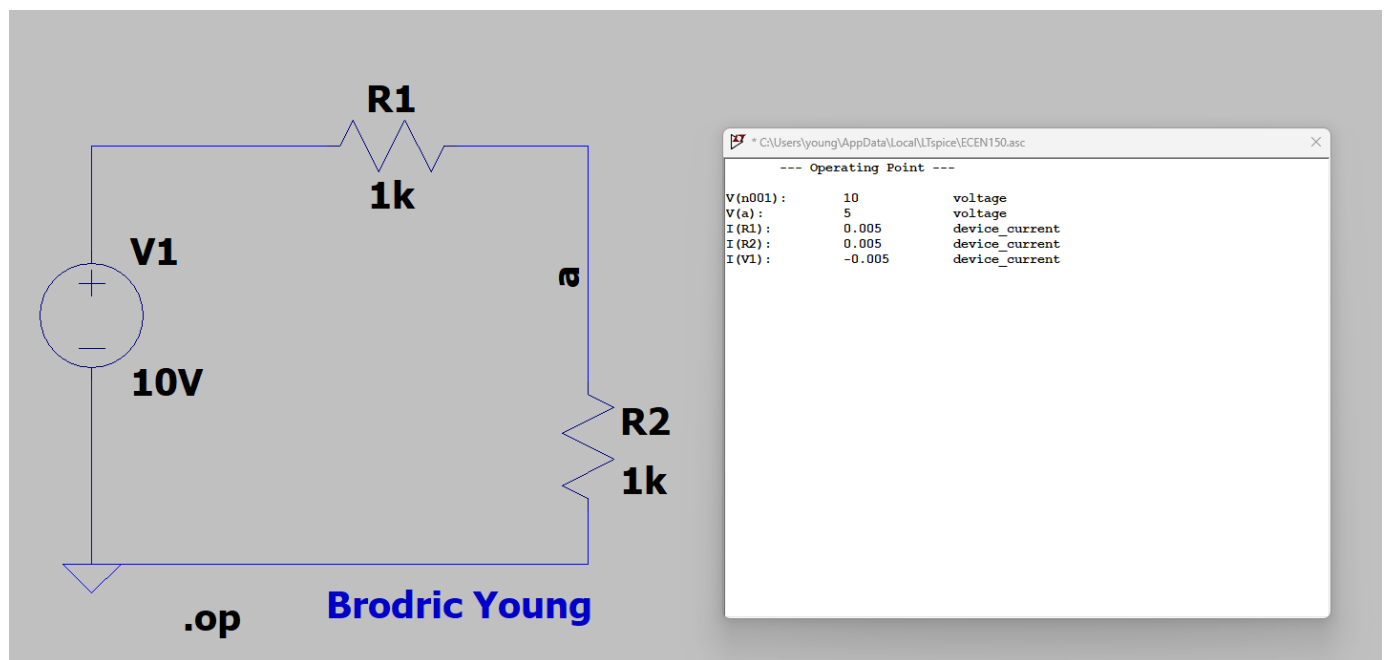


- .op Command: **Edit → Spice directive** (or  icon). Type “.op”, OK, then place.
- Delete: If you ever need to delete a wire, component, or label, click **Edit → Delete** (or the  icon), and then click on the item to be deleted.
- Be sure to save your work.

Step 2: Simulate the circuit.

- The “.op” command you placed instructs LTspice to simulate the DC operating point of the circuit, i.e., the DC values of all voltages and currents.
- To simulate: Click **Simulate → Run** (or the  icon)
- A window should pop up with the DC voltages and currents for each device and node.
 - Notes:
 - The source current is negative because it *leaves* the source.
 - Resistor current should always be positive because it is *absorbed*. However, the resistors in SPICE are polarity sensitive (unlike real ones).
 - If a resistor current is negative, it’s because the resistor is in “backwards”. Rotate it by clicking “move”, then “ctrl r” a few times.
- **Replace the figure below with your own. Ensure that your name is included** on the schematic and the simulation results are visible.

(Replace with a similar version of your own) **(10 points)**




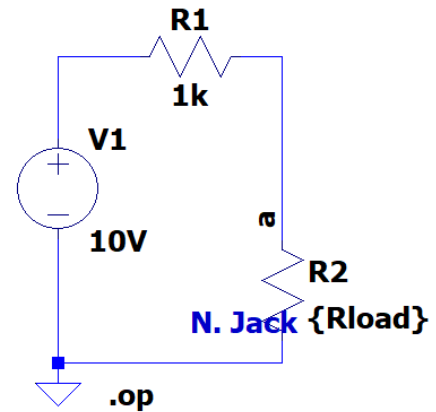
Question 1: Do the results match your expectations? Show the calculations for what Ohm’s Law predicts for the current and node voltage “a” in your answer. **(4 points)**

Yes it does match my expectations. $10V / 2000\text{ohms} = 0.005A$ so it is correct.
 $0.005A * 1000\text{ohms} = 5V$ for node a as well, just as expected.

Part 2. DC sweep simulation.


Step 1: Modify the circuit.

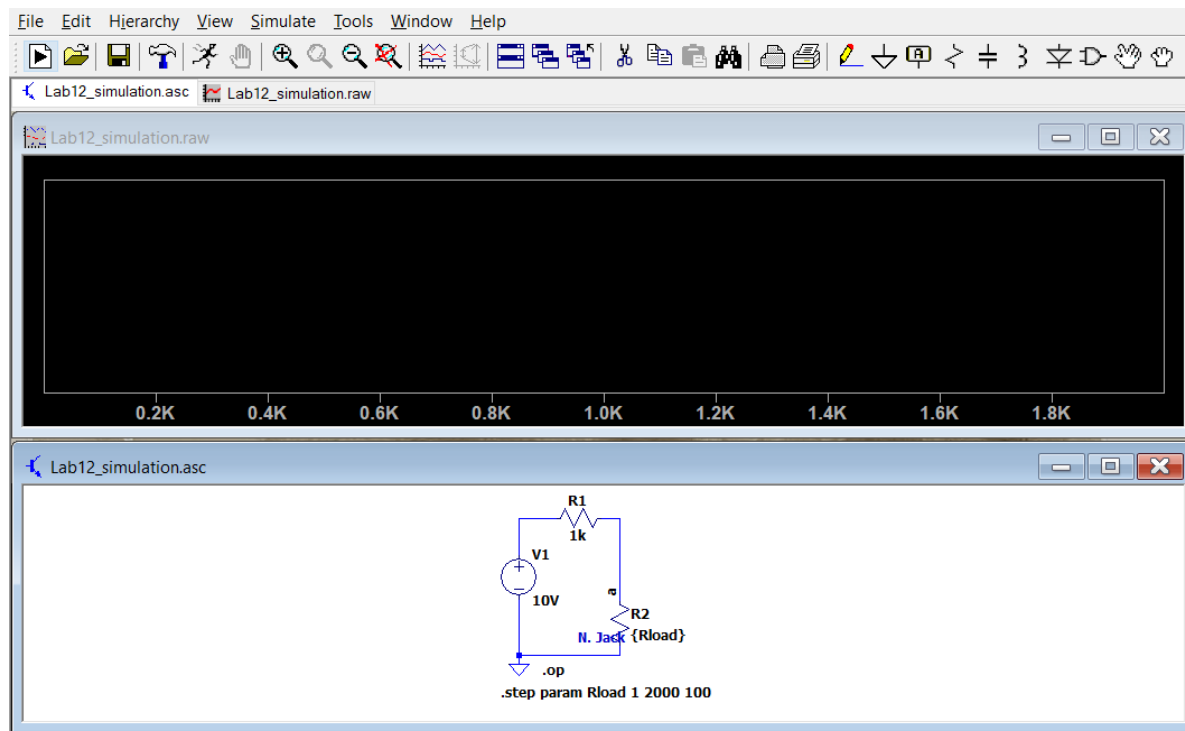
- Modify the circuit to look like the one here.
 - Change the value of R2 to be "{Rload}"
 - This defines the value to be a variable parameter rather than a specific number.
 - **.step**: Add a 2nd simulation directive **Edit** → **Spice directive** (or  icon)
 - **.step param Rload** means we will be stepping (or "sweeping") through several values of Rload in our simulation.
 - The "1 2000 100" indicates we will start with 1 Ω , step to 2000 Ω , in 100 Ω increments.
- Save your work.



.step param Rload 1 2000 100

Step 2: Run the parameter sweep simulation.

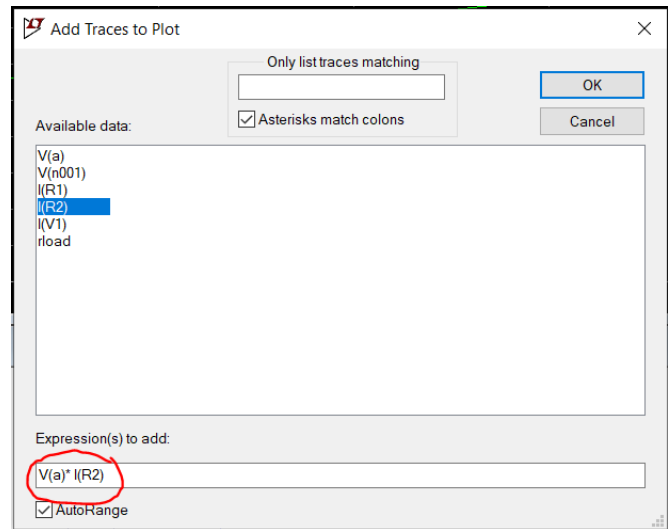
- **To simulate**: Click **Simulate** → **Run** (or the  icon)
- Your screen should look something like this:



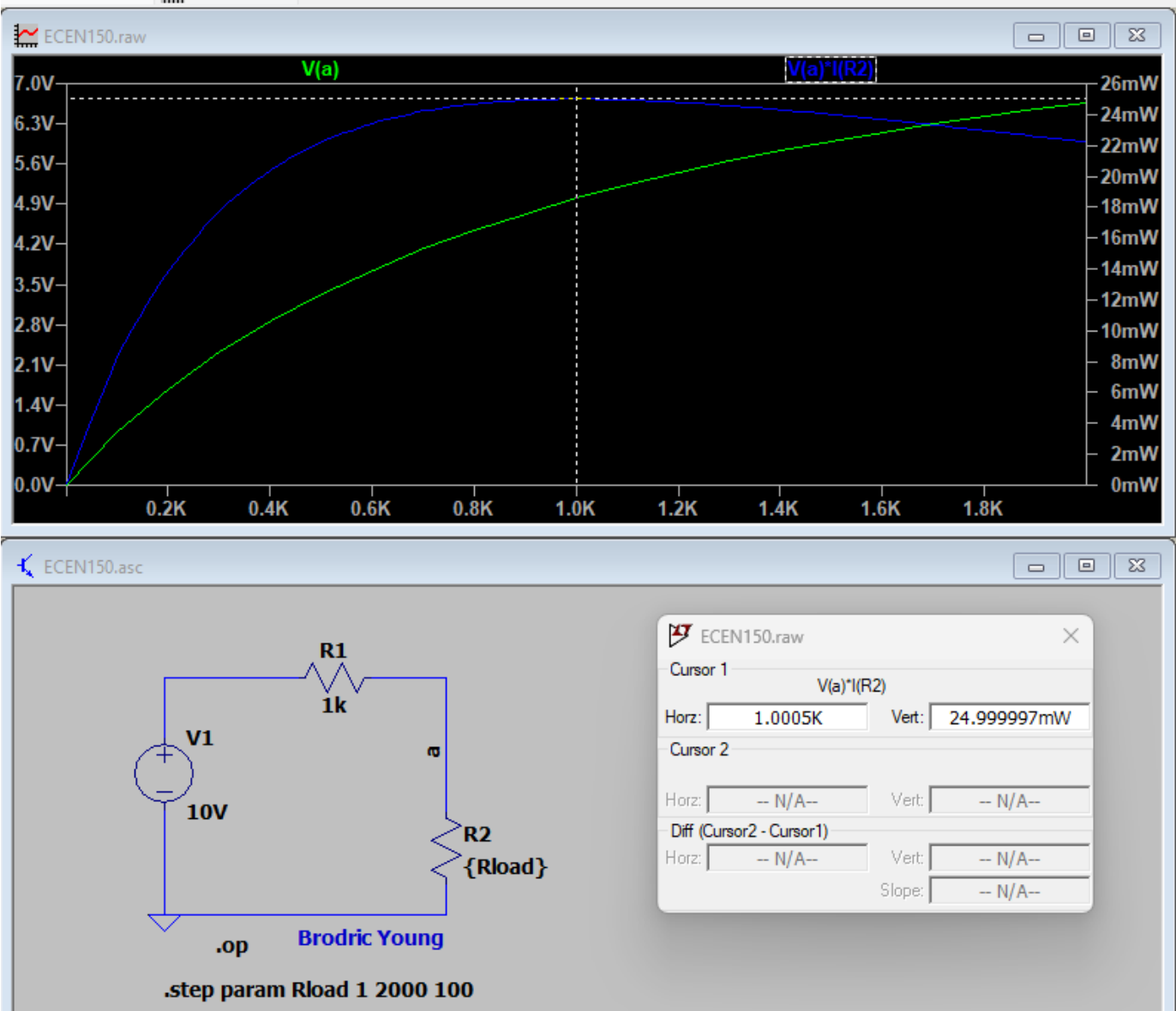
- **Right-click in the black window** and select "Add Traces". This allows you to choose the results (traces) to plot.
 - Select the "V(a)" signal, and then click "OK". This adds the voltage at node "a" to your plot.
 - Right-click → Add traces again. This time, create this expression at the bottom of the window: **V(a)*I(R2)** (See screen shot below). You can do this by clicking "V(a)", then typing the "*" after it, then clicking "I(R2)". Or, you can type the whole thing. Then click "OK".

- You should now see both waveforms plotted. **The x-axis is the Rload value.**
- Add a trace on the “power” plot:
 - Click on the colored label of that trace (i.e., the label that says “V(a)*I(R2)”)
 - This adds a cursor to your power waveform.
- Hover your mouse over the dashed cursor line.** Drag it until you find the maximum power (y-axis value). Make note of the x-axis (Rload) value as well.
- Replace the figure below with your own.**

Ensure that your name is included on the schematic, the plot and schematic are visible, and the cursor values are visible.



(Replace with a similar version of your own) (10 points)



Question 2: According to the maximum power transfer theorem, for what value of Rload should the power delivered to R2 be at a maximum? What should that maximum power value be? Show your calculations and compare against what you found in the simulation. (4 points)

(Enter your response here)

Rload should be the same as R, which would be 1k ohms to get the max power delivered to Rload. The value of the max power should be $(V^2) / R_{load} = (5V^2) / 1000\text{ohms} = 0.025W = 25mW$ which matches what I found in the simulation.

*****Demo your LTspice simulation to the TA; then take Lab 12: Quiz 1*****

Part 3. Conclusions statement.

Write a brief conclusions statement that addresses the questions below. Please use complete sentences and correct grammar to express your thoughts on how you fulfilled the purposes of the lab:

Questions:

- Why do you think LTspice would be useful for circuit design and analysis?
- What is the difference between a “.op” simulation and a “.step” simulation?
- Why might you need to use a “.step” simulation?

Conclusions (6 points):

LTspice would be useful for big complicated circuits and for testing different values of things because it's easy to build circuits and it can calculate what's going on for you as well as change the values for you. The “.op” simulation tells the program to simulate using the DC values for the voltages and currents in the schematic, while the “.step” simulation means you step through several values for something. In our case we stepped through values for {Rload} going from 1 to 2000 stepping by 100's . The “.step” simulation is very useful if you want to test different values for an element which can be used to create a graph which is easy to see things such as the max and min of whatever it is.

Congratulations, you have completed Lab!
You may now submit this document.